



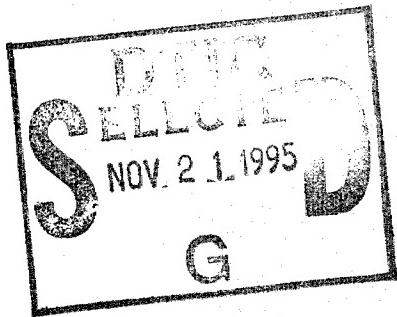
Technical Report REMR-HY-13
August 1995

**US Army Corps
of Engineers**
Waterways Experiment
Station

Repair, Evaluation, Maintenance, and Rehabilitation Research Program

HIVEL2D User's Manual

by R. C. Berger, Richard L. Stockstill, Mikel W. Ott



Approved For Public Release; Distribution Is Unlimited

19951116 101

DTIC QUALITY INSPECTED 5



Prepared for Headquarters, U.S. Army Corps of Engineers

The following two letters used as part of the number designating technical reports of research published under the Repair, Evaluation, Maintenance, and Rehabilitation (REMR) Research Program identify the problem area under which the report was prepared:

Problem Area		Problem Area	
CS	Concrete and Steel Structures	EM	Electrical and Mechanical
GT	Geotechnical	EI	Environmental Impacts
HY	Hydraulics	OM	Operations Management
CO	Coastal		

The contents of this report are not to be used for advertising, publication, or promotional purposes. Citation of trade names does not constitute an official endorsement or approval of the use of such commercial products.



PRINTED ON RECYCLED PAPER

HIVEL2D User's Manual

by R. C. Berger, Richard L. Stockstill, Mikel W. Ott

U.S. Army Corps of Engineers
Waterways Experiment Station
3909 Halls Ferry Road
Vicksburg, MS 39180-6199

Accesion For		
NTIS	CRA&I	<input checked="" type="checkbox"/>
DTIC	TAB	<input type="checkbox"/>
Unannounced <input type="checkbox"/>		
Justification		
By		
Distribution /		
Availability Codes		
Dist	Avail and / or Special	/
A-1		

Final report

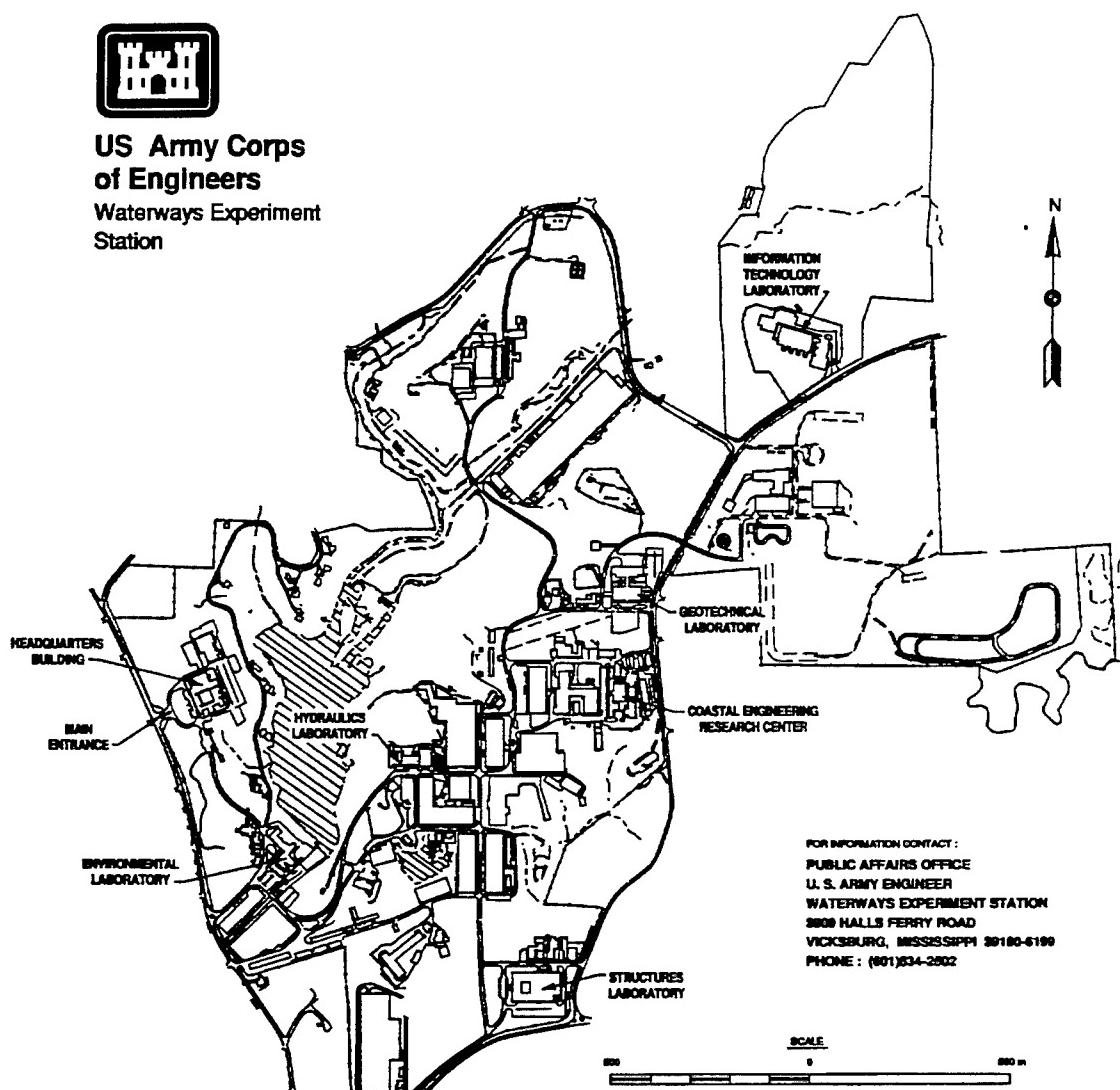
Approved for public release; distribution is unlimited

Prepared for U.S. Army Corps of Engineers
Washington, DC 20314-1000

Under Civil Works Research Work Unit 32657



**US Army Corps
of Engineers**
Waterways Experiment
Station



Waterways Experiment Station Cataloging-in-Publication Data

Berger, Rutherford C.

HIVEL2D user's manual / by R.C. Berger, Richard L. Stockstill, Mikel W. Ott ; prepared for U.S. Army Corps of Engineers.

40 p. : ill. ; 28 cm. — (Technical report ; REMR-HY-13)

Includes bibliographic references.

1. Streamflow — Mathematical models. 2. Hydraulic jump — Models. 3. Hydrodynamics — Data processing. 4. River channels. I. Stockstill, Richard L. II. Ott, Mikel W. III. United States. Army. Corps of Engineers. IV. U.S. Army Engineer Waterways Experiment Station. V. Repair, Evaluation, Maintenance, and Rehabilitation Research Program. VI. Title. VII. Series: Technical report (U.S. Army Engineer Waterways Experiment Station) ; REMR-HY-13.

TA7 W34 no.REMR-HY-13

Contents

Preface	iv
1—Introduction	1
Background	1
Purpose and Scope	1
2—HIVEL2D Overview	2
Governing Equations	2
Finite Element Model	4
Petrov-Galerkin Test Function	5
Shock Capturing	6
Temporal Derivatives	6
Solution of the Nonlinear Equations	7
Model Features	7
3—Developing an Application	9
Grid Generation	9
Hydrodynamic Input	10
Hot Start File	12
Running the Model	16
Output File	17
Viewing the Output	18
References	19
Appendix A: Boundary Conditions	A1
Flow In and Out	A1
Flow In, No Flow Out	A3
No Flow In and Flow Out	A3
No Flow In or Out	A4
Appendix B: Troubleshooting	B1
Appendix C: Example Problem	C1
Appendix D: Interpolation Program	D1
Program Description	D1
Interpolation Scheme for Quadrilateral Elements	D2

Interpolation Scheme for Triangular Elements	D3
Interpolation	D4

SF 298

Preface

The study reported herein was authorized by Headquarters, U.S. Army Corps of Engineers (HQUSACE), as part of the Hydraulics Problem Area of the Repair, Evaluation, Maintenance, and Rehabilitation (REMR) Research Program. The work was performed under Civil Works Research Work Unit 32657, "Model for Evaluation and Maintenance of High-Velocity Channels." The REMR Technical Monitor was Mr. Dave Wingerd (CECW-EH).

Mr. William N. Rushing (CERD-C) was the REMR Coordinator at the Directorate of Research and Development, HQUSACE. Mr. James E. Crews (CECW-O) and Dr. Tony C. Liu (CECW-EG) served as the REMR Overview Committee. The REMR Program Manager was Mr. William F. McCleese (CEWES-SC-A). Mr. Glenn A. Pickering, Chief, Hydraulic Structures Division (HSD), Hydraulics Laboratory (HL), U.S. Army Engineer Waterways Experiment Station (WES), was the Problem Area Leader. Dr. R. C. Berger, Estuaries Division (ED), HL, and Mr. Richard L. Stockstill, HSD, were Principal Investigators.

This work was conducted by Dr. Berger, Mr. Stockstill, and Mr. Mikel W. Ott, HSD, under the direct supervision of Mr. Pickering and under the general supervision of Messrs. Frank A. Herrmann, Jr., Director, HL; Richard A. Sager, Assistant Director, HL; William H. McAnally, Jr., Chief, ED; and John F. George, Chief, Locks and Conduits Branch, HSD, HL.

At the time of publication of this report, Director of WES was Dr. Robert W. Whalin. Commander was COL Bruce K. Howard, EN.

The contents of this report are not to be used for advertising, publication, or promotional purposes. Citation of trade names does not constitute an official endorsement or approval of the use of such commercial products.

1 Introduction

Background

HIVEL2D is a free-surface, depth-averaged, two-dimensional model designed specifically for flow fields that contain supercritical and subcritical regimes as well as the transitions between the regimes. The model provides numerically stable solutions of advection-dominated flow fields containing shocks such as oblique standing waves and hydraulic jumps.

HIVEL2D has been verified by comparing computed model results with laboratory data. The findings of these tests are presented in Stockstill and Berger (1994).

Purpose and Scope

This report serves as a user's manual for HIVEL2D Version 1.07. First, a brief description of the model equations are presented. Next, step-by-step instructions are given to guide the user in creating the required input files. These input files include the geometry (numerical model computational mesh) file, the hydrodynamic parameters/boundary conditions file, and the initial conditions file. Finally, the numerical model output file is explained. Appendix A describes the boundary conditions. Appendix B describes problems with the model and solutions. Appendix C gives an example problem. Appendix D discusses the interpolation program.

2 HIVEL2D Overview

HIVEL2D (Stockstill and Berger 1994) is designed to simulate flow typical in high-velocity channels. The model is a finite element description of the two-dimensional shallow-water equations in conservative form. The model does not include Coriolis or wind effects as these are typically not important in high-velocity channels.

Governing Equations

Vertical integration of the equations of mass and momentum conservation for incompressible flow with the assumption that vertical accelerations are negligible compared to the acceleration of gravity results in the governing equations commonly referred to as the shallow-water equations. The dependent variables of the two-dimensional fluid motion are defined by the flow depth h , the volumetric discharge per unit width in the x-direction p , and the volumetric discharge per unit width in the y-direction q . These variables are functions of the independent variables x and y , the two space directions, and time t . The shallow-water equations in conservative form are given as (Abbott 1979):

$$\frac{\partial \mathbf{Q}}{\partial t} + \frac{\partial \mathbf{F}_x}{\partial x} + \frac{\partial \mathbf{F}_y}{\partial y} + \mathbf{H} = \mathbf{0} \quad (1)$$

where

$$\mathbf{Q} = \begin{pmatrix} h \\ p \\ q \end{pmatrix} \quad (2)$$

$$\mathbf{F}_x = \begin{pmatrix} p \\ \frac{p^2}{h} + \frac{1}{2}gh^2 - \frac{h\sigma_{xx}}{\rho} \\ \frac{pq}{h} - \frac{h\sigma_{yx}}{\rho} \end{pmatrix} \quad (3)$$

$$\mathbf{F}_y = \begin{pmatrix} q \\ \frac{pq}{h} - \frac{h\sigma_{xy}}{\rho} \\ \frac{q^2}{h} + \frac{1}{2}gh^2 - \frac{h\sigma_{yy}}{\rho} \end{pmatrix} \quad (4)$$

$$\mathbf{H} = \begin{pmatrix} 0 \\ gh\frac{\partial z_o}{\partial x} + g\frac{n^2 p \sqrt{p^2+q^2}}{C_0^2 h^{7/3}} \\ gh\frac{\partial z_o}{\partial y} + g\frac{n^2 q \sqrt{p^2+q^2}}{C_0^2 h^{7/3}} \end{pmatrix} \quad (5)$$

where

$p = uh$, u being the depth-averaged x-direction component of velocity

$q = vh$, v being the depth-averaged y-direction component of velocity

g = acceleration due to gravity

$\sigma_{xx}, \sigma_{xy}, \sigma_{yx}, \sigma_{yy}$ = Reynolds stresses where the first subscript indicates the direction and the second indicates the axis direction normal to the face on which the stress acts

ρ = fluid density

z_o = channel invert elevation

n = Manning's roughness coefficient

C_0 = dimensional constant ($C_0 = 1$ for SI units and $C_0 = 1.486$ for non-SI units)

The Reynolds stresses are determined using the Boussinesq approach relating stress to the gradient in the mean currents:

$$\begin{aligned}\sigma_{xx} &= 2\rho v_t \frac{\partial u}{\partial x} \\ \sigma_{xy} = \sigma_{yx} &= \rho v_t \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \\ \sigma_{yy} &= 2\rho v_t \frac{\partial v}{\partial y}\end{aligned}\quad (6)$$

where v_t is the eddy viscosity, which varies spatially and is solved empirically as a function of local flow variables (Rodi 1980):

$$v_t = \frac{Cn\sqrt{8g}\sqrt{p^2 + q^2}}{C_0 h^{1/6}} \quad (7)$$

where C is a coefficient that varies between 0.1 and 1.0.

The typical high-velocity channel is rather wide and shallow, so that the bulk of the turbulence is bed-generated and therefore Equation 7 is reasonable. However, if the walls or other structures make a significant contribution to the overall turbulence, this equation will underpredict the magnitude of the eddy viscosity or turbulence. Even under typical channel configurations there is fairly large uncertainty in the parameter C . This is apparent since it has been found to range from 0.1 to 1.0. Therefore, the user should consider making a sensitivity check on the calculated water surface for a range of values of C and any other parameters having values that are largely uncertain.

Finite Element Model

This system of partial differential equations is solved using the finite element method. The finite element approach taken is a Petrov-Galerkin formulation that incorporates a combination of the Galerkin test function and a non-Galerkin component to control oscillations due to convection. An integration by parts procedure is used to develop the weak form of the equations. The weak form facilitates the specification of boundary conditions. It is given as:

$$\sum_e \left[\int_{\Omega_e} \left(\psi_i \frac{\partial Q}{\partial t} - \frac{\partial \phi_i}{\partial x} F_x - \frac{\partial \phi_i}{\partial y} F_y + \varphi A \frac{\partial Q}{\partial x} + \varphi B \frac{\partial Q}{\partial y} + \psi_i H \right) d\Omega_e + \oint_{\Gamma_e} \phi_i (F_x n_x + F_y n_y) dl \right] = 0 \quad (8)$$

where the variables are understood to be discrete values and

e = subscript indicating a particular element

Ω = domain

$\psi_i = \phi I + \varphi_i$ = test function

ϕ_i = Galerkin part of the test function

I = identity matrix

φ_i = non-Galerkin part of the test function

$(n_x, n_y) = \hat{n}$ = unit vector outward normal to the boundary Γ_e

and

$$A = \frac{\partial F_x}{\partial Q} \quad (9)$$

$$B = \frac{\partial F_y}{\partial Q}$$

Natural boundary conditions are applied to the sidewall boundaries through the weak form. The sidewall boundaries are “no-flux” boundaries; that is, there is no net flux of mass or momentum through these boundaries. These boundary conditions are enforced through the line integral in the weak form.

Petrov-Galerkin Test Function

The Petrov-Galerkin test function ψ_i is defined (Berger 1993) as:

$$\psi_i = \phi I + \varphi_i \quad (10)$$

where

$$\varphi_i = \beta \left(\Delta x \frac{\partial \phi_i}{\partial x} \hat{A} + \Delta y \frac{\partial \phi_i}{\partial y} \hat{B} \right) \quad (11)$$

where β is a dissipation coefficient varying in value from 0 to 0.5, the ϕ terms are the linear basis functions, and Δx and Δy are the grid intervals. A detailed explanation of this test function, in particular \hat{A} and \hat{B} , is given in Berger (1993).

Shock Capturing

The coefficient β scales the dissipation needed for numerical stability. More dissipation is needed in the vicinity of shocks such as hydraulic jumps than in smooth regions of the flow field. Because a lower value of β ($\beta = \beta_{SM}$) is more precise, a large value of β ($\beta = \beta_{SH} = 0.5$ where β_{SM} and β_{SH} are the Petrov-Galerkin parameters for smooth flow and for shocks, respectively) is applied only in regions in which it is needed. HIVEL2D employs a mechanism that detects shocks and increases β automatically. Therefore, β_{SH} is implemented only when needed as determined by evaluation of the element energy deviation. In a similar manner, the eddy viscosity coefficient C varies from C_{SM} to C_{SH} , the effect being that eddy viscosity is increased only in areas of greatest element energy deviation.

Temporal Derivatives

A finite difference expression is used for the temporal derivatives. The general expression for the temporal derivative of a variable Q_j is:

$$\begin{aligned} \left(\frac{\partial Q_j}{\partial t} \right)^{m+1} &= \left(\frac{1 + \alpha}{2} \right) \left(\frac{Q_j^{m+1} - Q_j^m}{t^{m+1} - t^m} \right) \\ &\quad + \left(\frac{1 - \alpha}{2} \right) \left(\frac{Q_j^m - Q_j^{m-1}}{t^m - t^{m-1}} \right) \end{aligned} \quad (12)$$

where j is the nodal location and m is the time-step. An α equal to 1 results in a first-order backward difference approximation, and an α equal to 2 results

in a second-order backward difference approximation of the temporal derivative.

Solution of the Nonlinear Equations

The system of nonlinear equations is solved using the Newton-Raphson iterative method. Let R_i be a vector of the nonlinear equations computed using a particular test function ψ_i and using an assumed value of Q_j . R_i is the residual error for a particular test function i . Subsequently, R_i is forced toward zero as:

$$\frac{\partial R_i^k}{\partial Q_j^k} \Delta Q_j^k = -R_i^k \quad (13)$$

where the derivatives composing the Jacobian are determined analytically and k is the iteration number. This system of equations is solved for ΔQ_j^k and then an improved estimate for Q_j^{k+1} is obtained from:

$$Q_j^{k+1} = Q_j^k + \Delta Q_j^k \quad (14)$$

This procedure is continued until convergence to an acceptable residual error is obtained.

Model Features

Particular features of HİVEL2D Version 1.07 are as follows:

- a. Combinations of linear-based triangular and rectilinear shape functions are used to represent p , q , and h .
- b. The model is compiled to solve problems having meshes as large as 2000 nodes and 2000 elements and will run on a 386-based (or higher) IBM-compatible personal computer having 8 MB of RAM.
- c. A Newton-Raphson iteration is used within each time-step.
- d. The mild slope assumption is invoked. This means the slope should be less than about 0.05. This assumes that the slope is geometrically mild, not hydraulically mild.

- e. Boundary conditions are specified for any combination of supercritical and subcritical inflow and outflow.
- f. Boundary conditions are constant over the simulation period.
- g. Boundary roughness (Manning's n) is specified on an element type basis.
- h. A user-specified parameter α produces either first-order or second-order backward temporal differences.
- i. A Petrov-Galerkin approach in which the test function is weighted upwind along characteristics is employed. The degree of upwinding and thus stability is determined by the parameters β_{SM} and β_{SH} . Defaults are $\beta_{SM} = 0.25$ and $\beta_{SH} = 0.5$.
- j. A shock detection mechanism based upon the energy variation per element is used to invoke β_{SH} .
- k. Turbulent eddy viscosity is calculated based upon simple user-specified parameters C_{SM} and C_{SH} using velocity, depth, and roughness.

3 Developing an Application

The basic steps to developing an application of HIVEL2D are as follows:

- a. Generate the grid:
 - (1) Number the grid intelligently.
 - (2) Identify inflows and outflows.
- b. Develop a hydrodynamic input file.
- c. Develop a reasonable hot start file.
- d. Run the model (probably several times). If necessary, refine the grid and interpolate a new hot start file using INTRPL8.EXE (Appendix D).
- e. Run HIVELBIN.EXE on the output file to generate a solution file that can be viewed in FastTABS.
- f. Examine the solution for reasonableness.

Grid Generation

To a large degree the quality of the grid determines the accuracy and stability of the model. The first step in grid generation is getting the necessary geometry information into the grid generator. Critical elements such as points on transitions, bridge piers, and curves should be put into the grid generator. One method involves drawing the structure to be modeled in three dimensions in a CAD program, and then extracting the critical points. These critical points are then put into a grid generator. One grid generator the Corps of Engineers uses is FastTABS (Brigham Young University 1993). FastTABS is a pre- and postprocessor for programs involving two-dimensional finite element meshes. It was developed by the Engineering Computer Graphics Laboratory at Brigham Young University for the U.S. Army Engineer Waterways Experiment Station.

Remember the following general rules when generating the grid:

- a. HIVEL2D uses linear elements. Therefore, when generating a HIVEL2D grid, use only four-node quadrilaterals and three-node triangles.
- b. Keep the element aspect ratio less than 3:1 (the closer to 1:1, the better). The aspect ratio is the ratio of the longest element dimension to the shortest, i.e., the length-to-width ratio.
- c. Use gradual transitions in element size. Generally, an element's area should not be greater than 1½ times its smallest neighbor.
- d. Include at least five or six elements across a channel in the area of interest. If, for example, the channel has an island in the center, this resolution is needed on both sides of the island. Also, increase the resolution around grade breaks and wall transitions.

Once the grid generation is complete, be sure to renumber the grid using FastTABS' renumber options. The best numbering scheme will give the smallest bandwidth. Normally, the nodes need to be numbered progressively across the narrowest dimension of the grid. This minimizes the bandwidth, which makes HIVEL2D run more efficiently.

Before exiting the grid generator, note the nodes that compose the inflow and outflow boundaries. These nodes must be put into the geometry (grid) file. Open the geometry file with an ASCII text editor, such as the MS-DOS editor EDIT.COM. Add the BI lines for each node in which an inflow condition is to be input. Add a BO line for each outflow node (this will not always require a boundary condition). These lines are no longer formatted in v.1.07 of HIVEL2D. Be sure to type *BO*, not *B0* (zero). Keep in mind that the model is expecting a relatively narrow (computationally) dimension across the channel. The various lines can appear in any order. Any prefixes on each line that are not required by HIVEL2D are printed to the screen, but are otherwise ignored by the program.

Table 1 shows the geometry file format. If the geometry was generated using FastTABS, then the GNN and GE lines are transparent to the user. To generate a grid from scratch, be sure to follow the correct format shown in Table 1.

Hydrodynamic Input

Starting the model is usually the most difficult task. The model should be started with small time-steps (i.e., a CFL number less than 1) where:

Table 1
Geometry File Format

Prefix (Columns 1-3)	Column Read As..	Contents
Bl or bi	Free format	Node on the boundary in which flow enters the model
BO or bo	Free format	Node on the boundary in which flow leaves the model
GNN or gnn ¹	Free format	Node number, node's x-coordinate, node's y-coordinate, node's z-coordinate (invert elevation)
GE or ge ²	4-8	Element number
	9-14	Node number 1 of element number GE
	15-20	Node number 2 of element number GE
	21-26	Node number 3 of element number GE
	27-32	Node number 4 of element number GE (0 if element is a triangle)
	56-61	<u>Material type</u> - used to designate the bed roughness area of common type. The actual value of Manning's n is input in the hydrodynamic input file. The user can specify maximum of three material types.

¹ Need one GNN line for each node. The GNN's do not have to be in order, but no node numbers can be left out between 1 and the largest node number used.
² Need one GE line for each element giving the list of nodes associated with this element in counterclockwise order.

$$CFL = \frac{|U| + \sqrt{gh}}{\Delta l / \Delta t} \quad (15)$$

where

$$|U| = \sqrt{u^2 + v^2}$$

and

Δl = the element length

Δt = the time-step size

The time-step can then be gradually increased. If the steady-state solution is desired, then a fairly large time-step can be used. If the interest is in unsteady results, then, for accuracy, the CFL number should be held to a maximum of 2.

Appropriate boundary conditions must be supplied. For an inflow boundary condition when the flow is supercritical, the user must specify both x- and y-components of flow along with the depth. If the flow is subcritical here, then only the x- and y-components of flow are read. If the flow is subcritical at the outflow boundary, then the tailwater elevation must be specified. If the flow is supercritical at the outflow, then no boundary conditions need to be given. This is done in the input file by specifying a tailwater elevation that is lower than the bed elevation at the outflow boundary.

Starting the model is sometimes difficult when the flow is supposed to be supercritical at the outflow boundary. The model attempts to converge to a solution, but it may not be the desired solution. One way to avoid this is to start the model with a tailwater that is slightly subcritical (if the starting conditions are not known). Then after the model settles down, the boundary condition can be changed to supercritical. This takes a little experience.

Users should note that the Manning's n applies to each element type as well as the adjoining sidewalls. It is also used in conjunction with velocity, depth, and C to determine a turbulent eddy viscosity estimate.

The input to the hydrodynamics file is made through the list in Table 2. All variables are real except those beginning with I , J , K , L , M , or N , which are integer variables. All input is free-field. Each variable is described in Table 2 in the order in which it appears in the hydrodynamic input file. Units are designated by L and T , for length and time, respectively. If no units are specified, the variable is dimensionless.

Hot Start File

This file is the last two time-steps of information from the previous model run. Since the temporal derivative can be a second-order backward difference, two time-steps of old information are needed. If this is the initial run, then this information will have to be generated based upon experience. One method is to start the model with zero velocity and a constant depth. The data are read as free-field.

This file will be overwritten by the model run, so if there is a chance that this information will be needed later, save a copy. If the model run finishes but the data are bad for some reason, the hot start will be bad as well. Again, this is a good reason to save a copy.

Table 2
Hydrodynamic Input File Format

Line Number	Variables	Description
1	g	Gravitational acceleration (REAL), L/T^2
2	C_o^2 (C2)	C_o^2 (REAL) is the empirical conversion coefficient used in Manning's equation. This needs to be consistent with the units used for g . Typical values are 2.208 for English units and 1.0 for SI.
3	C_{SM} , C_{SH} (CSB,CSBSHK)	These coefficients (REAL) supply the turbulence coefficients in the normal flow region and near the jump, respectively. $0.1 \leq C_{SM}, C_{SH} \leq 1.0$ Typically, C_{SM} is chosen as 0.1 and C_{SH} is 0.5. These parameters determine the turbulent eddy viscosity based upon the depth, velocity, and roughness. Larger values imply larger turbulent eddy viscosity. Again, if the flow is relatively smooth and there are no jumps, set C_{SM} to the value used for C_{SH} .
4	Δt , α (DELT, ALPHAT)	<p>Δt (REAL) is the time-step size T (units need to be consistent with other variables). α (REAL) determines the precision of the temporal term in the model.</p> <p style="text-align: center;">$1.0 \leq \alpha \leq 2.0$</p> <p>$\alpha = 1.0$ for first-order backward difference $\alpha = 2.0$ for second-order backward difference</p> <p>On steady-state problems use 1.0 (it is more stable). On time-dependent problems generally use 2.0, but if the time-step is small it does not matter.</p>
5	NSTEPS, NRVL	<p>NSTEPS (INTEGER) is the number of time-steps to model in this run. NRVL (INTEGER) is the interval for which all hydrodynamic information is written to the hydrodynamic output file, which is for plotting. Small values for this can cause the output file to become <u>quite large</u>. If interest is only on the last time-step, set NRVL=NSTEPS.</p> <p>For example, entering a value of 2 will save time-steps 2, 4, 6,...</p>

(Sheet 1 of 3)

Table 2 (Continued)

Line Number	Variables	Description
6	ITMAX, TOL	<p>ITMAX (INTEGER) is the maximum number of Newton-Raphson iterations per time-step. A typical value is 4. TOL (REAL) is the convergence criteria. The number of Newton-Raphson iterations per time-step is ITMAX unless convergence is reached first. A typical value is 0.005.</p> <p>If $\Delta F < \text{TOL}$, then the solution is considered converged and no more iterations are performed for the current time-step:</p> <p>Where</p> <p>$\Delta F = \text{maximum of } \{ (F - F_0) / F \text{ (for each node)}$</p> <p>$F$ = Froude number this iteration F_0 = Froude number last iteration</p>
7	IUORP _i , ND _i , VX _i , VY _i , H _i	<p>These are the inflow input lines. One line is supplied for each inflow node (these are the nodes identified by BI in the geometry file).</p> <p>ND_i (INTEGER) is the individual node number. VX_i, VY_i (REALS) are the components of either velocity (u, v), L/T, or unit discharge (p, q), L^2/T, at this node. The type is determined by IUORP_i (Table 3). H_i (REAL), L, is the node depth. This is used for supercritical inflow boundary conditions. H_i is used or ignored depending on IUORP_i (Table 3). It is not necessary to put a value here if H_i is to be ignored.</p>
8	TAIL	TAIL (REAL) is the tailwater elevation, L, (this will be assigned to all nodes that have been designated as outflow nodes, BO lines, in the geometry file). If this elevation is less than the bed elevation at this node, the model assumes that the flow is supercritical there and no boundary condition is assigned.
9	NMTYP	NMTYP (INTEGER) is the number of element types for roughness. This list begins with type 1 and does not skip an integer. These element types are assigned in the geometry file.

(Sheet 2 of 3)

Table 2 (Concluded)

Line Number	Variables	Description
10	j, FRIC _j	j (INTEGER) is the element type. FRIC _j (REAL) is the Manning's n roughness for element type j. There should be NMTYP number of these lines, beginning with j=1 through j=NMTYP, in order, with no skips. This friction applies to the bed and the sidewalls of the elements of this type.
<p>Note: The parameters β_{SM} and β_{SH} (Petrov-Galerkin weight coefficients) are not required in the hydrodynamic input file. The defaults are $\beta_{SM} = 0.25$ and $\beta_{SH} = 0.5$. These values can be overridden by including as the first line of the hydrodynamic input (for example):</p> <p>OR 0.20 0.5</p> <p>where OR is the flag that instructs the program to read the following numbers as $\beta_{SM} = 0.20$ and $\beta_{SH} = 0.5$</p>		
(Sheet 3 of 3)		

Table 3
Designation of Boundary Condition Type

Flow Regime	Specified Variables	IUORP
Supercritical	p, q, h	2
Supercritical	u, v, h	-2
Subcritical	p, q	1
Subcritical	u, v	-1

The information in the hotstart file is:

(Line 1) TIME

(Line 2) $p^m, q^m, h^m, p^{m-1}, q^{m-1}, h^{m-1}$ (for each node, in order)

where

TIME = the time associated with values at time-step m

m = the last time-step

m-1 = the next to the last time-step

Remember that this file is free format.

The initial hot start file can also cause starting a run to be tricky. If the interest is in steady-state conditions, the accuracy of the initial guess will determine how long it will take to reach steady state. If the geometry is simple, then there is no trouble in guessing the discharge components. However, if it is complicated, it may be necessary to resort to zero discharge and a specified depth, which is a typical method. If the walls are sloping, there may be some difficulty in specifying the depth as well. It is fairly simple to write a program to subtract the bed elevation from some set of water-surface elevations. Typically, the user should set the old and the even older time-step information in this file as identical.

Running the Model

To run the model the user must supply four file names, three input and one output. Note the following example (the computer prompts are in bold and the user's commands and responses are in italics):

C:\>*HIVEL2D*

WHAT IS THE GEOMETRY FILE ?

example.geo

**WHAT IS THE HYDRODYNAMIC AND COMPUTATION
PARAMETER INPUT FILE ?**

example.flo

WHAT IS THE PLOT OUTPUT FILE ?

example.out

**WHAT IS THE HYDRODYNAMIC OUTPUT FILE (THIS IS
ALSO THE NAME OF THE INITIAL DATA FILE FOR
HOTSTART) ?**

example.hot

- a. The executable is called HIVEL2D.EXE.
- b. Each file name can be up to 15 characters for Unix and Apple machines, and 12 characters for IBM compatibles (e.g., EXAMPLE1.GEO).
- c. The file extensions of GEO, FLO, OUT, and HOT are ones that are typically used to designate files, but these suffixes are arbitrary.
- d. The geometry file contains the node locations, the element connection table, and the designation of boundary inflow and outflow nodes.

- e. The hydrodynamic and computation parameter input file contains the hydraulic information about the run as well as the computation parameters. Examples of computation parameters are the specific boundary values of depth and flows, roughness, time-step size, number of iterations, etc.
- f. The output file contains the water-surface elevation and velocity fields for all nodes at each time location designated in the hydrodynamic input file. This information can then be converted to a form that can be viewed in a postprocessor such as FastTABS.
- g. The hot start file contains the time, p , q , and depth for the last two time-steps. This is used to start the calculation. This file is overwritten by each successful run; so, if it might be needed later, make a copy of it under another name before running the program.

After the file names are input, the program will begin running and several banners will appear on the screen.

- a. First, the program informs the user of any unused lines in the geometry file and prints the contents of the unused lines.
- b. Next, the contents of the hydrodynamic input file are displayed on the screen to allow the user to check the values that were read.
- c. From the initial conditions the average energy, the minimum vorticity and the element where it occurred, and the maximum vorticity and the element where it occurred are calculated and displayed.
- d. If the coefficients C_{SM} , C_{SH} , or α are out of range, the program will report the error and halt.
- e. Then, as the program runs, the results for each time-step are displayed. These results include the number of iterations required, the maximum residual error and the node with which it is associated, the average energy, the minimum vorticity and the element where it occurred, and the maximum vorticity and the element where it occurred.

Output File

This file is output from the HIVEL2D run and is intended mainly to supply information for plotting in FastTABS. The output file contains the following:

- a. The number of time-steps saved, number of elements, and number of nodes.
- b. The time.

- c. Velocity components of u and v , and water-surface elevation for each node.
- d. Material type number for each element.

Note: these are repeated for each time-step that was output.

The number of time-steps saved is determined by the request in the hydrodynamic input file for calculated time-steps and NRVL (the interval to save time-steps to this file). This file can get quite large, so assign NRVL with discretion.

Viewing the Output

The output files generated by HVEL2D are ASCII text files, which, with some manipulation, can be imported to almost any numerical postprocessor available. The preferred postprocessor is FastTABS, because the output files are in a format that can be directly converted to the appropriate binary form required by FastTABS. The program that converts HVEL2D output files to FastTABS binary solution files is called HVELBIN. The required version of HVELBIN depends on the FastTABS version used.

References

- Abbott, M. B. (1979). *Computational hydraulics, elements of the theory of free surface flows*. Pitman Advanced Publishing Limited, London.
- Berger, R. C. (1993). "A finite element scheme for shock capturing," Technical Report HL-93-12, U.S. Army Engineer Waterways Experiment Station, Vicksburg, MS.
- Brigham Young University. (1993). "FastTABS 2.2 MS-Windows Version," Engineering Computer Graphics Laboratory, Brigham Young University, Provo, UT.
- Rodi, W. (1980). "Turbulence models and their application in hydraulics - A state of the art review," State-of-the Art Paper, International Association for Hydraulic Research, Delft, The Netherlands.
- Stockstill, R. L., and Berger, R. C. (1994). "HIVEL2D: A two-dimensional flow model for high-velocity channels," Technical Report REMR-HY-12, U.S. Army Engineer Waterways Experiment Station, Vicksburg, MS.

Appendix A

Boundary Conditions

Flow In and Out

Case 1

Flow regime

Supercritical upstream
Supercritical downstream

Boundary conditions

p, q, h specified upstream (defined in Chapter 2)

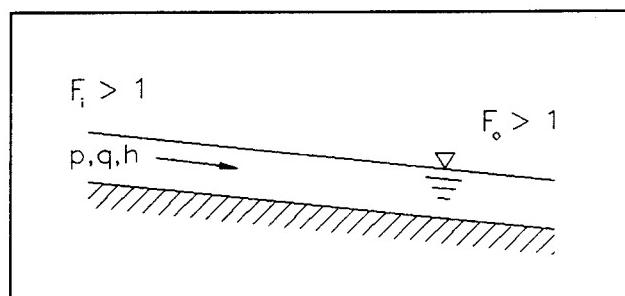


Figure A1. Case 1 sketch

Comments

It can be difficult to get the outflow boundary to converge at startup. One method is to specify a tailwater such that the flow is barely subcritical, then after some time, when the resulting jump is nearly gone, remove the downstream boundary condition. Note: F_i = inflow Froude number; F_o = outflow Froude number.

Case 2

Flow regime

Supercritical upstream
Subcritical downstream

Boundary conditions

p, q, h specified upstream
 h specified downstream

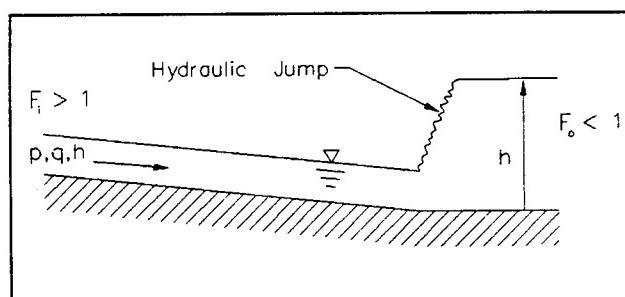


Figure A2. Case 2 sketch

Case 3

Flow regime
Subcritical upstream
Supercritical downstream

Boundary conditions
 p, q specified upstream

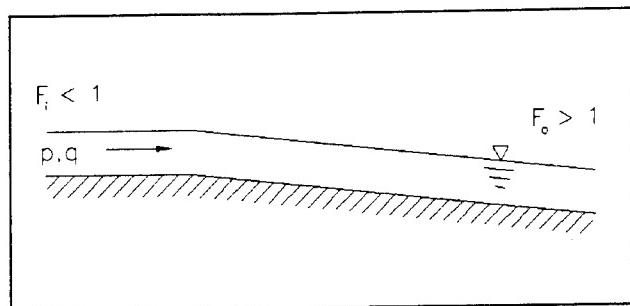


Figure A3. Case 3 sketch

Case 4

Flow regime
Subcritical upstream
Subcritical downstream

Boundary conditions
 p, q specified upstream
 h specified downstream

Comments

This problem may take a long time to reach steady state.

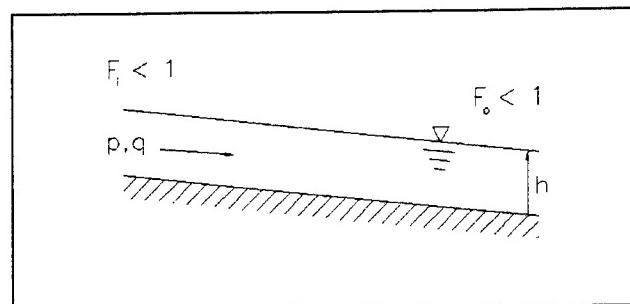


Figure A4. Case 4 sketch

Case 5

Flow regime
Subcritical upstream
Subcritical downstream

Boundary conditions
 p, q specified upstream
 h specified downstream

Comments

Even though this contains a steep section in which the flow is supercritical, the boundary conditions are the same as for Case 4. The tailwater specified downstream causes the hydraulic jump.

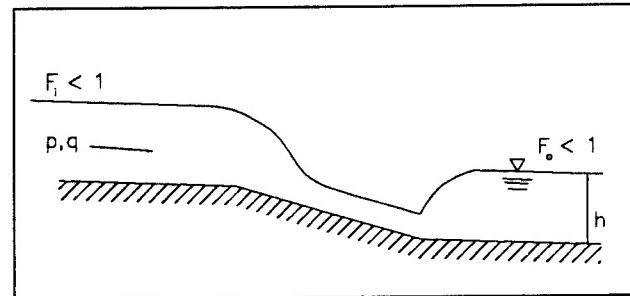


Figure A5. Case 5 sketch

Flow In, No Flow Out

Case 6

Flow regime

Supercritical upstream
Wall downstream

Boundary conditions

p, q, h upstream
Leave out BO lines
(BI and BO defined in
Table 1)

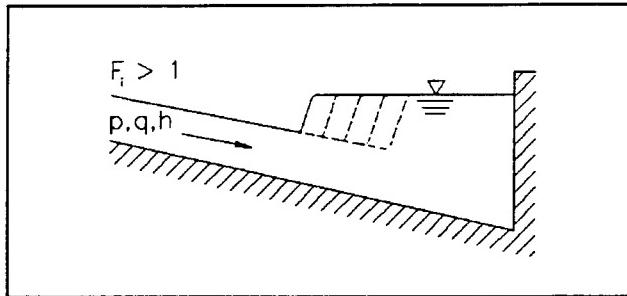


Figure A6. Case 6 sketch

Comments

No steady state, keeps filling. Note the moving jump that forms.

Case 7

Flow regime

Subcritical upstream
Wall downstream

Boundary Conditions

p, q specified upstream
Leave out BO lines in
geo file

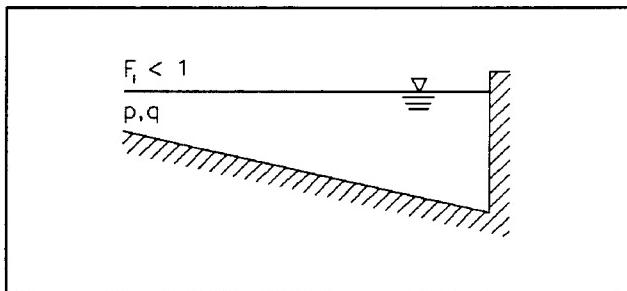


Figure A7. Case 7 sketch

No Flow In, Flow Out

Case 8

Flow regime

Wall upstream
Supercritical downstream

Boundary conditions

Leave out BI lines

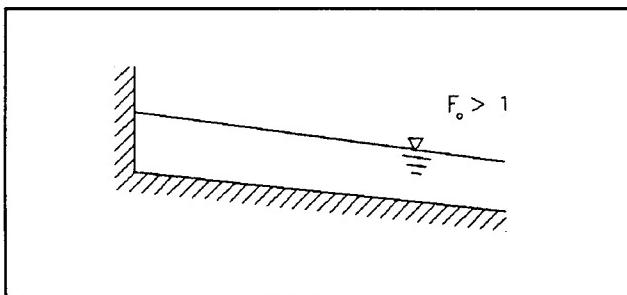


Figure A8. Case 8 sketch

Comments

No steady state, goes dry. This is probably a very hard problem to run.

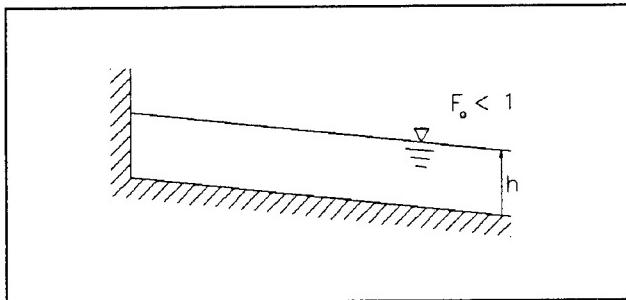
Case 9

Flow regime

Wall upstream
Subcritical downstream

Boundary conditions

Leave out BI lines
Specify h downstream



Comments

No steady state, goes dry. This is also a very hard problem to run.

Figure A9. Case 9 sketch

No Flow In or Out

Case 10

Flow regime

Wall upstream
Wall downstream

Boundary conditions

Leave out BI and BO lines

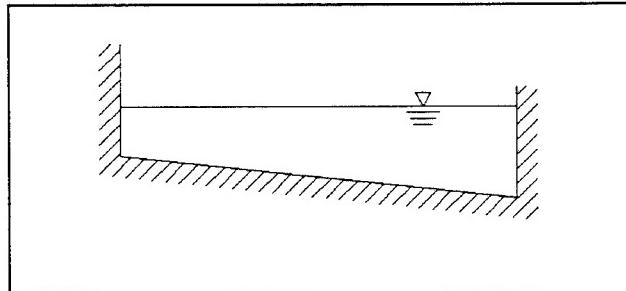


Figure A10. Case 10 sketch

Appendix B

Troubleshooting

Table B1
Problems and Solutions

Problem	Possible Solutions
1. Model won't converge	Try more iterations per time-step. Reduce time-step. If lower boundary is migrating away from solution: First set a tailwater, wait until jump is nearly gone, then remove tailwater boundary condition. Call for help: be prepared to send the geometry, hydraulic input, output, and hot start files.
2. Model keeps filling up (water level continuously rises).	Make sure you didn't leave out BO lines in geometry file. Use BO and not BO (zero) (Table 1).
3. Converges to a bizarre solution.	Make sure resolution is adequate.
4. Problems reading hot start file.	Verify that the time is included as the first line of the file. Verify the number of lines. May have to put a carriage return at the end of the last line of the hot start file.

Appendix C

Example Problem

The example grid is shown in Figure C1. The flow enters on the left and exits to the right. The 2,000-ft (610-m)-long model is supercritical at both boundaries, so all information is specified at the upstream end. The example contains a transition from a 100-ft (30.5-m) width to a 90-ft (27.4-m) width. The slope is 0.01 except in the 100-ft (30.5-m) length of transition, where it is 0.02. The transition is to have a bed roughness higher than the rest of the model. The model includes a 50-ft (15.2-m)-long by 18-ft (5.5-m)-wide bridge pier. Note that the resolution is concentrated around and downstream of any abrupt changes (Figure C2). Since this flow is supercritical, all information including shocks and rough conditions is swept downstream. Therefore, this is where the resolution is needed.

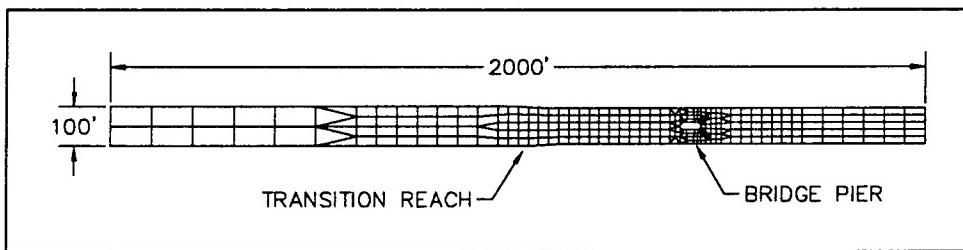


Figure C1. Complete example grid and geometry

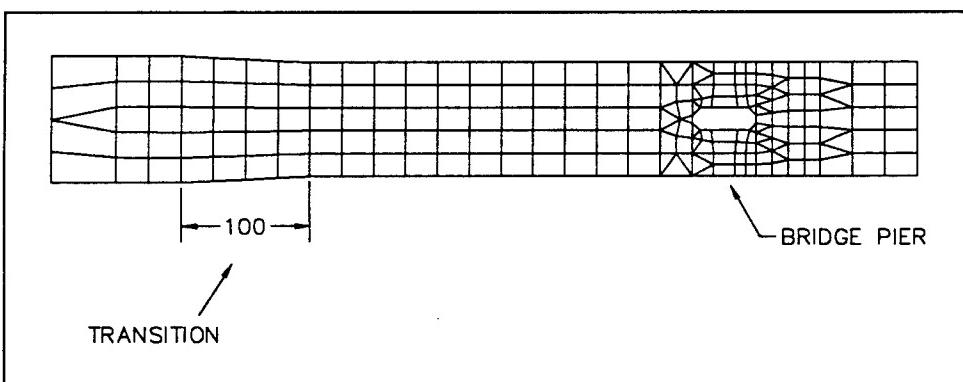


Figure C2. Detail of example grid showing transition and bridge pier

The complete geometry file includes 338 nodes and 302 elements. The nodes 1, 2, and 3 are the upstream boundary nodes (designated by BI in the geometry file), and nodes 333-338 are the downstream boundary nodes (designated by BO in the geometry file). An abbreviated version of the geometry file is shown in Figure C3.

```

T1 example geometry file, contains 338 nodes, representing a bridge pier
T2 and a contraction, the contraction is steeper and rougher than
T3 the rest of the model
GO 1 2 3 -1
BI 1
BI 2
BI 3
BO 333
BO 334
BO 335
BO 336
BO 337
BO 338
GE 1 5 2 1 4 0 0 0 0 1 0.0
GE 2 6 3 2 5 0 0 0 0 1 0.0
GE 3 8 5 4 7 0 0 0 0 1 0.0
GE 4 9 6 5 8 0 0 0 0 1 0.0
GE 5 11 8 7 10 0 0 0 0 1 0.0
GE 6 12 9 8 11 0 0 0 0 1 0.0
GE . .
GE . .
GE 299 335 329 328 334 0 0 0 0 1 0.0
GE 300 336 330 329 335 0 0 0 0 1 0.0
GE 301 337 331 330 336 0 0 0 0 1 0.0
GE 302 338 332 331 337 0 0 0 0 1 0.0
GNN 1 0.0000 0.0000 0.0000
GNN 2 0.0000 50.0000 0.0000
GNN 3 0.0000 100.0000 0.0000
GNN 4 100.0000 0.0000 -1.0000
GNN 5 100.0000 50.0000 -1.0000
GNN 6 100.0000 100.0000 -1.0000
GNN . .
GNN . .
GNN 334 2000.0000 23.0000 -20.0000
GNN 335 2000.0000 41.0000 -20.0000
GNN 336 2000.0000 59.0000 -20.0000
GNN 337 2000.0000 77.0000 -20.0000
GNN 338 2000.0000 95.0000 -20.0000

```

Figure C3. Example geometry file (example.geo)

The initial hot start file can be tricky. This case involves a straight reach so it is fairly easy to set up a file with p , q , h of 60., 0., 3. as a constant throughout the domain. If the domain is curved one might have to start with pooled flow at a single depth. The beginning of the initial hot start file is shown in Figure C4.

The initial input file is shown in Figure C5.

The text in the hydrodynamic input file that describes each numerical input is not actually read by the program so it can be left out or changed. The initial time-step was chosen to produce a CFL number (Equation 15) of roughly 1 (so this time-step is chosen as 1.0 sec). The temporal derivative

```
0.000
60.  0.  3.  60.  0.  3.
60.  0.  3.  60.  0.  3.
60.  0.  3.  60.  0.  3.
60.  0.  3.  60.  0.  3.
60.  0.  3.  60.  0.  3.
60.  0.  3.  60.  0.  3.
60.  0.  3.  60.  0.  3.
60.  0.  3.  60.  0.  3.
.
.
.
(This line is repeated at least 338 times, one line for each node)
```

Figure C4. Example hot start file (example.hot).

```

32.2                      GRAVITY (FT/(SEC*SEC))
2.208                      MANNING CONVERSION
0.1   0.2                  CB FOR VISCOSITY SMOOTH, AND JUMP
1.0   1.0      TIME STEP SIZE, TEMPORAL DIFF 1.=FIRST 2.= SECOND ORDER
20 20                      TIME STEPS, OUTPUT INTERVAL
4     0.005    NUMB. OF ITERATIONS, CONVERGENCE CRITERIA
2     1       60.  0. 3.0  TYPE BOUNDARY, BC NODE, P,Q,H, (when required)
2     2       60.  0.   3.0
2     3       60.  0.   3.0
-14.8                      TAILWATER ELEVATION
2                           NUMBER OF ELEMENT TYPES (FOR ROUGHNESS)
1     0.015
2     0.025

```

Figure C5. Initial hydrodynamic input file (example1.flo).

was chosen to be first-order since the interest is in steady-state results only. Even though the downstream boundary is intended to be supercritical and no boundary condition is needed, experience has shown that the model sometimes will converge to a different solution or be unstable unless a very good starting guess at the solution was made. Therefore, until the model settles down, a tailwater elevation is specified corresponding to flow that is slightly subcritical. Two different element types and Manning's roughness coefficients have been designated. The convergent section is steeper and rougher in this example. The model will run 20 time-steps resulting in a simulation time of 20 sec. The time-step size is then increased. Later the downstream tailwater condition is effectively removed by specifying an elevation that is below the bed elevation.

The next input file is run to raise the time-step now that the model has settled down from startup. The time-step has been raised to 3.0 sec. The second input file is shown in Figure C6.

The final input file (Figure C7) removes the tailwater constraint. This run was repeated one time.

Figures C8 and C9 are the steady-state results of the model run for water-surface and velocity contours, respectively. These were generated by converting the output plot file to a binary file in the form expected for a solution by FastTABS. There are many options available such as vectors and time-series plots. The results show that the bridge pier is a choke, so that the flow is subcritical in front of the pier. This is apparent since the shock contour is

```

32.2          GRAVITY (FT/ (SEC*SEC))
2.208          MANNING CONVERSION
0.1  0.2      CB FOR VISCOSITY SMOOTH, AND JUMP
3.0  1.0      TIME STEP SIZE, TEMPORAL DIFF 1.=FIRST 2.=SECOND ORDER
20 20          TIME STEPS, OUTPUT INTERVAL
4      0.005    NUMB. OF ITERATIONS, CONVERGENCE CRITERIA
2      1      60.  0.   3.0  TYPE BOUNDARY, BC NODE, P,Q,H, (when required)
2      2      60.  0.   3.0
2      3      60.  0.   3.0
-14.8          TAILWATER ELEVATION
2          NUMBER OF ELEMENT TYPES (FOR ROUGHNESS)
1      0.015
2      0.025

```

Figure C6. Second hydrodynamic input file (example2.flo).

```

32.2          GRAVITY (FT/ (SEC*SEC))
2.208          MANNING CONVERSION
0.1  0.2      CB FOR VISCOSITY SMOOTH, AND JUMP
1.0  1.0      TIME STEP SIZE, TEMPORAL DIFF 1.=FIRST 2.=SECOND ORDER
20 20          TIME STEPS, OUTPUT INTERVAL
4      0.005    NUMB. OF ITERATIONS, CONVERGENCE CRITERIA
2      1      60.  0.   3.0  TYPE BOUNDARY, BC NODE, P,Q,H, (when required)
2      2      60.  0.   3.0
2      3      60.  0.   3.0
-30.          TAILWATER ELEVATION
2          NUMBER OF ELEMENT TYPES (FOR ROUGHNESS)
1      0.015
2      0.025

```

Figure C7. Final hydrodynamic input file (example3.flo).

perpendicular to the flow instead of being swept back. Note also that the flow returns to supercritical flow along the pier sides and a depression wave has the sweptback wake.

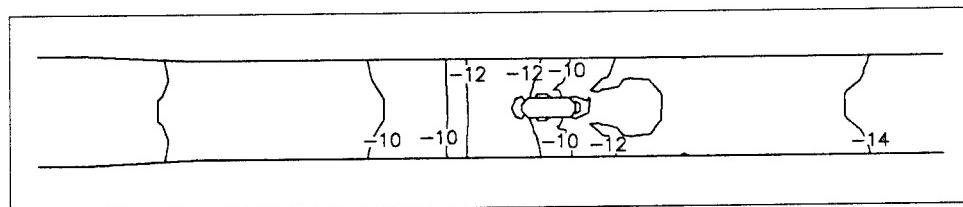


Figure C8. Final water-surface contours, ft

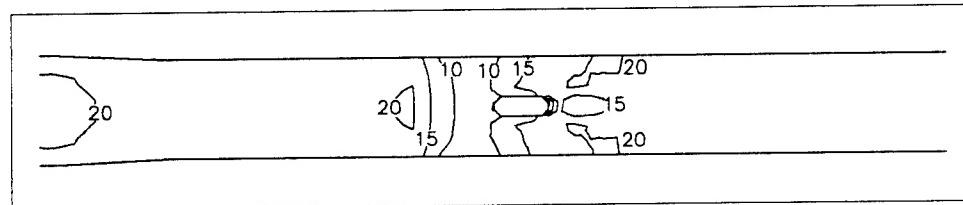


Figure C9. Final velocity contours, fps

Appendix D

Interpolation Program

The interpolation program INTRPL8.EXE allows the user to refine a grid for which answers (p , q , and h , defined in Chapter 2) have already been found. Using the old hot start information, the old geometry, and the new geometry, the user can generate a new hot start file for the new geometry. This new hot start file is then the starting point for a new run of HLEVEL2D.

The program also outputs an interpolated plot output file that can be converted to binary and viewed in FastTABS. This is used to verify the interpolated values. This feature can also be used to interpolate from a fine grid to a coarse grid, to better view vectors on a very fine grid, for example. Users should note that the program cannot interpolate answers for any nodes that lie outside the old geometry's boundary. For instance, refining a mesh around a curve will cause the nodes along the curve's outer radius to lie outside the old coarse mesh's boundary. The program will report the nodes that lie outside the domain.

Program Description

After input of the necessary data, the first routine the program performs is that of comparing the old grid with the new grid. Old node numbers are typically going to be different than new ones. For this reason, the old answers at each node are first "mapped" onto the corresponding nodes on the new grid by comparing the coordinates of the nodes.

Next comes the task of looking at each node, seeing if it is a new node, and, if it is a new node, finding out in which old element it is located. There are two possible locations for a new or "target" node: in a quadrilateral element or in a triangular element. A node on an element side is interpolated in the element in which it is first found.

Deciding in which element a target node is located is done by a cross product scheme, in which the cross product is taken between a vector lying on the side of an element and a vector from a corner node on that side to the target

point. If the cross products for all sides are greater than zero, then the node lies within that element.

Interpolation Scheme for Quadrilateral Elements

For convenience in interpolating, a local coordinate system (ξ, η) is used where $-1 \leq \xi \leq 1$ and $-1 \leq \eta \leq 1$ (Figure D1) to derive the bilinear interpolation functions. First the target point (x_p, y_p) is written in (ξ, η) coordinates.

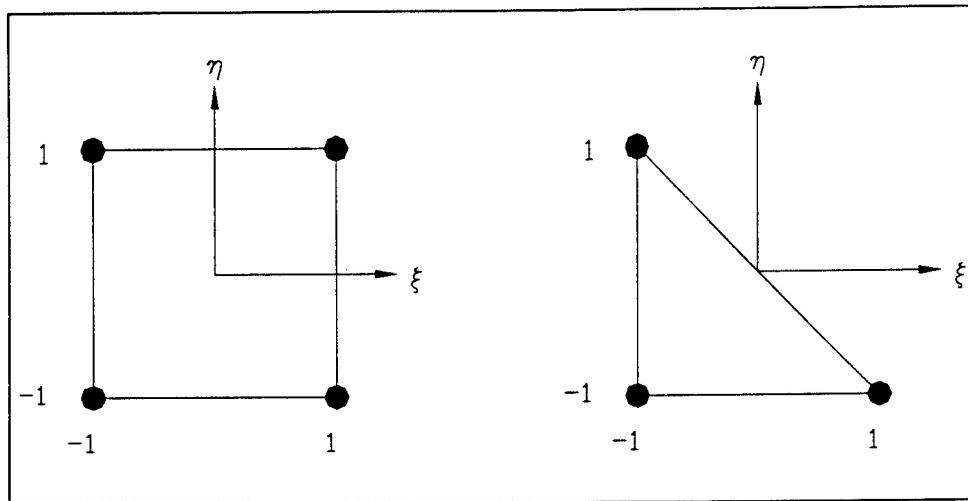


Figure D1. Local coordinates for quadrilateral and triangular elements

This is done using Newton-Raphson iteration to solve the following equations:

$$x = \sum_{i=1}^4 \phi_i x_i \quad (D1)$$

and

$$y = \sum_{i=1}^4 \phi_i y_i \quad (D2)$$

for the local coordinates ξ and η where the bilinear shape functions for a quadrilateral element are:

$$\phi_1 = \frac{1}{4}(1-\xi)(1-\eta)$$

$$\phi_2 = \frac{1}{4}(1+\xi)(1-\eta)$$

$$\phi_3 = \frac{1}{4}(1+\xi)(1+\eta)$$

$$\phi_4 = \frac{1}{4}(1-\xi)(1+\eta)$$

Interpolation Scheme for Triangular Elements

Again, the local coordinate system (ξ, η) where $-1 \leq \xi \leq 1$ and $-1 \leq \eta \leq 1$, (Figure D1) is used to derive the interpolation functions. Solving

$$x = \sum_{i=1}^3 \phi_i x_i \quad (D3)$$

and

$$y = \sum_{i=1}^3 \phi_i y_i \quad (D4)$$

simultaneously for ξ and η , where:

$$\phi_1 = -\frac{1}{2}(\xi + \eta)$$

$$\phi_2 = \frac{1}{2}(1 + \xi)$$

$$\phi_3 = \frac{1}{2}(1 + \eta)$$

are the bilinear shape functions for a triangular element, results in the following equations for ξ and η :

$$\xi = -\frac{\beta_1}{\alpha_1} + \frac{\alpha_2(-\alpha_3\beta_1 + \alpha_1\beta_2)}{\alpha_1(-\gamma_1 + \gamma_2 + \gamma_3 - \gamma_4 - \gamma_5 + \gamma_6)} \quad (D5)$$

and

$$\eta = - \frac{(-\alpha_3\beta_1 + \alpha_1\beta_2)}{(-\gamma_1 + \gamma_2 + \gamma_3 - \gamma_4 - \gamma_5 + \gamma_6)} \quad (D6)$$

where

$$\alpha_1 = x_1 - x_2$$

$$\alpha_2 = x_1 - x_3$$

$$\alpha_3 = y_1 - y_2$$

$$\beta_1 = 2x_p - x_2 - x_3$$

$$\beta_2 = 2y_p - y_2 - y_3$$

$$\gamma_1 = x_2 y_1$$

$$\gamma_2 = x_3 y_1$$

$$\gamma_3 = x_1 y_2$$

$$\gamma_4 = x_3 y_2$$

$$\gamma_5 = x_1 y_3$$

$$\gamma_6 = x_2 y_3$$

Interpolation

Once the target node has been located, and mapped onto old grid element local coordinates ξ and η , the interpolated values of p , q , and h are found using the following equations:

$$h = \sum_{i=1}^n \phi_i h_i \quad (D7)$$

$$p = \sum_{i=1}^n \phi_i p_i \quad (D8)$$

$$q = \sum_{i=1}^n \phi_i q_i \quad (D9)$$

where n is the number of nodes on the old grid element and ϕ is the appropriate interpolation function.

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188
<p>Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.</p>			
1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE August 1995	3. REPORT TYPE AND DATES COVERED Final report	
4. TITLE AND SUBTITLE HIVEL2D User's Manual		5. FUNDING NUMBERS WU 32657	
6. AUTHOR(S) R. C. Berger Richard L. Stockstill Mikel W. Ott			
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) U.S. Army Engineer Waterways Experiment Station 3909 Halls Ferry Road, Vicksburg, MS 39180-6199		8. PERFORMING ORGANIZATION REPORT NUMBER Technical Report REMR-HY-13	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) U.S. Army Corps of Engineers, Washington, DC 20314-1000		10. SPONSORING/MONITORING AGENCY REPORT NUMBER	
11. SUPPLEMENTARY NOTES Available from National Technical Information Service, 5285 Port Royal Road, Springfield, VA 22161.			
12a. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release; distribution is unlimited.		12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) HIVEL2D is a numerical flow model for the evaluation of high-velocity channels. The depth-averaged, two-dimensional model is designed to address flow conditions commonly found in high-velocity channels. High-velocity channels are characterized by flow fields containing supercritical and subcritical flow and transitions from one regime to the other. This report serves as a user's manual for HIVEL2D Version 1.07. Included is a brief description of the model equations followed by step-by-step instructions for creating the model input files. An example problem is included.			
14. SUBJECT TERMS Finite element method High-velocity channels Hydraulic jump			15. NUMBER OF PAGES 40
			16. PRICE CODE
17. SECURITY CLASSIFICATION OF REPORT UNCLASSIFIED	18. SECURITY CLASSIFICATION OF THIS PAGE UNCLASSIFIED	19. SECURITY CLASSIFICATION OF ABSTRACT UNCLASSIFIED	20. LIMITATION OF ABSTRACT